

LIFT AND DRAG PERFORMANCE OF NACA0012 AIRFOIL AT VARIOUS ANGLE OF ATTACK USING CFD

GANDHI MALLELA¹, PALLAVI PATURU² & M. KOMALESWARAO³

^{1,2}Assistant Professor, Department of Mechanical Engineering, Vel Tech Rangarajan Dr. Sagunthala R&D
Institute of Science and Technology, Chennai, Tamil Nadu, India

³Assistant Professor, Department of Mechanical Engineering, JNTUK - University College of Engineering,
Vizianagaram, Andhra Pradesh, India

ABSTRACT

In this paper, an expansion of my research work ^[1] we complete a numerical investigation of lift and drag performance of the NACA0012 airfoil at a different negative and positive angle utilizing computational fluid dynamic. This investigation of the two dimensional (2D) subsonic fluid flow over an airfoil at an alternate approach extending from negative angle - 6 degree to positive point 10 degrees (distinction of 2 degrees AoA) at consistent Mach number is available. For this testing and investigation, we utilized a model of K- ω SST turbulence. The final product of the analysis, computational fluid dynamic reenactment demonstrates the applicability to the energizing finding the literature. Through this testing, we are communicating a dependable option test technique for discovering coefficient of drag and lift.

KEYWORDS: Turbulence, AoA, Lift, Drag & CFD

Received: Mar 16, 2018; **Accepted:** Apr 06, 2018; **Published:** Apr 25, 2018; **Paper Id.:** IJMPERDJUN201810

INTRODUCTION

It is a reality of general expertise that body in moving through a fluid covering with a resultant force that by and hugged in a very primarily movement of the resistance. A category of the body exists, regardless, that the fragment of the resultant force ordinarily to the orientation of the event is sometimes additional clear than the contradicting the event to boost the likelihood of the flight of a plane depends upon the usage of the body of this category for wing structure.

The approach is that the purpose between the approach air or relative breeze and a reference line on the plane or wings as shown in figure 1. As this nose of the wing turns up, approach increments and raise a force in addition increased. Drag goes up, however, additionally not as fast as a raise. Within the interior of activity a briefing creates a specific speed and after the pilot flips the plane, that's the pilot controls with the controls that the nose of the plane return up and, at some approach, the wings create enough to raise to bring the plane into the air.

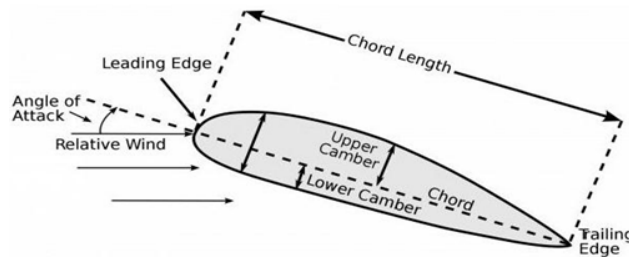


Figure 1: Regular Surface of Aerofoil Nomenclature

METHODOLOGY

Geometrical Model

NACA0012 symmetric airfoil geometry was picked up as arranging vertices i.e. Writings, content, records and imported into the ANSYS FLUENT. Barely any progressions were made to this to cure the geometry and make it significant as a CFD illustrate. Recognizable is fundamental amid the time spent doing CFD analysis; it makes the working environment where the dissent is emulated. A fundamental part of this is making the work incorporating the inquiry. This ought to be extended in all over to get the physical properties of the including fluid for this circumstance moving air. The work and angle ought to in like manner be assembled remembering the ultimate objective to characterize as far as possible conditions effectively.

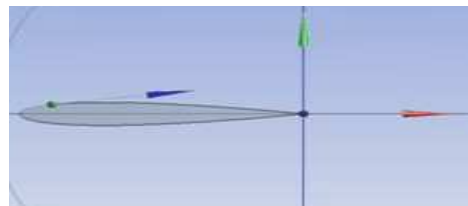


Figure 2: Geometrical Construction Model of NACA Airfoil

Immediately we have to import the directions of the airfoil and make the bend, the 2D analysis sort is used and dispatch the outline model made. By then we need to make the surface of the bend, at that point the airfoil is created. We need to make a sorted out system at the tail of the airfoil, this will empower us to influence the geometry of the C-mesh a workspace by using sketcher instruments. Next, we need to influence a surface from this to depict. The last progress of influencing the C-mesh work making a surface between the point of confinement and the airfoil by using boolean activities. In the last progress of making the geometry, we will set up the new surface into four quadrants; this will be significant for a cross section of the geometry. The geometry is finished. Save the assignment and close the arrangement modeler, as we are currently we are prepared to make the work for the reproduction.

Meshing and Boundary Conditions

An area including 2 squares and 1 crescent incorporates the NACA0012 symmetrical airfoil. The mesh is produced to be fine as local people nears the airfoil and with high essentials and coarser, more separation a long way from the airfoil. For this airfoil a shorted out quadratic mesh was used. In light of limitations in the FLUENT programming, the work must be fine also in particular areas quite a while to wrap up. For the NACA airfoil, the grid spread with an extending partition between center points, starting from little sizes from the primary edge. With the end goal of the maximum thickness of the airfoil to the trailing edge, huge measure of focus is dispersed on the airfoil surface.

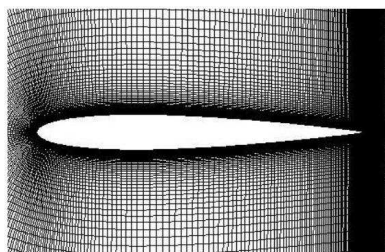


Figure 3: Mesh Generated around a NACA0012 Airfoil

Table 1: Operating Parameters of Input Values

Name of the Input	Value's
Flow Velocity (V)	Mach 0.44
Temperature (T)	300 Kelvin
Pressure (P)	0.0 (Pascal)
Model of Turbulence	K- ω SST
Density of Fluid (ρ)	01.225 kg/m ³
Fluid Viscosity (ν)	01.79 x E ⁻⁰⁰⁵ kg/m-s
Length (L)	1 unit
AOA (in degrees)	-6,-4,-2,0, 2,4,6,8 and 10
Fluid	Air as ideal gas

RESULTS

In this investigation numerical performed. The investigation was driven at subsonic turbulent stream Mach numbers is 0.44, twist speed around a NACA0012 airfoil at different points in the region of - 6 and 10 degrees were estimations is reenacted utilizing the turbulence show and the obtained result about are contrasted and the literature review of trial information. The varieties of lift coefficient and drag coefficient versus the AOA are given in figures, separately. The estimations of CL and CD for various AoA as got through FLUENT are displayed underneath in forbidden configuration.

Table 2: The Values of Cl and Cd for Different AOA

Angles of Attack ($^{\circ}$)	Coefficient of Lift	Coefficient of Drag
-6	-6.97×10^{-1}	1.26×10^{-2}
-4	-4.72×10^{-1}	9.09×10^{-3}
-2	-2.32×10^{-1}	8.96×10^{-3}
0	-1.66×10^{-5}	8.12×10^{-3}
2	2.30×10^{-1}	8.94×10^{-3}
4	4.72×10^{-1}	9.09×10^{-3}
6	6.99×10^{-1}	1.25×10^{-2}
8	9.17×10^{-1}	1.68×10^{-4}
10	$1 \times 10^{+00}$	3.12×10^{-2}

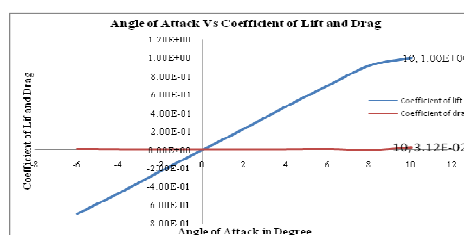
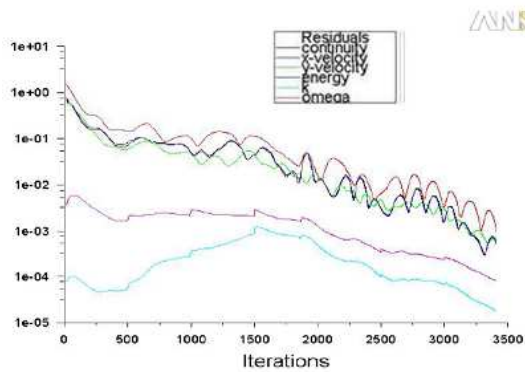
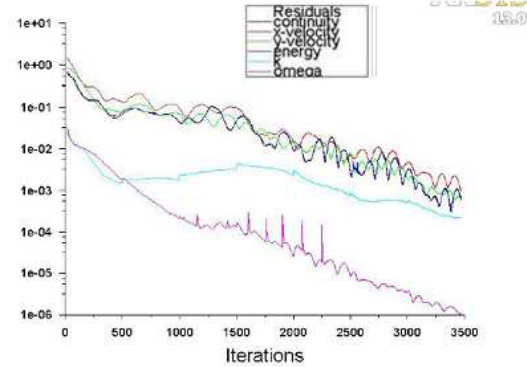


Figure 4: Angle of Attack Vs Coefficient of Lift and Drag at -6 to 10 Degrees

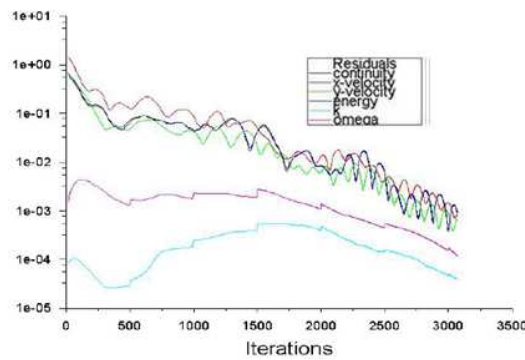
After post -processing in FLUENT following residual, Cd, Cl and Cp plots, Pressure and velocity contours were generated. According to calculated data from various angles of attack ranging from -6 degree to 10 degrees with an interval of 2 degrees. Iterations are generated at a constant of 2500 to 3000.



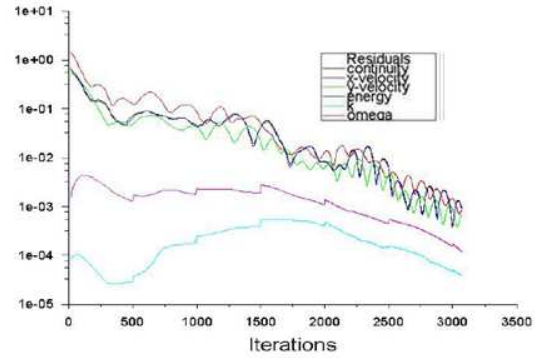
(a) -6 Degree of AoA



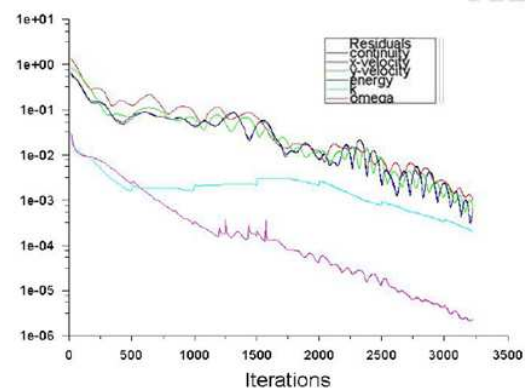
(b) -4 Degree of AoA



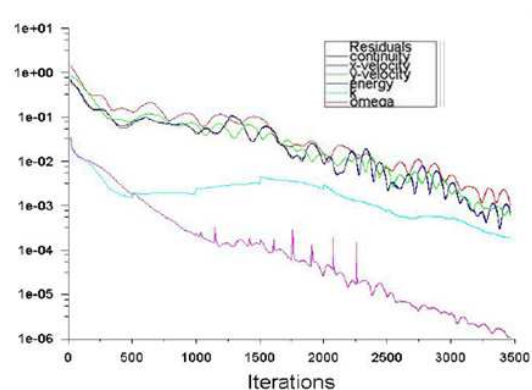
(c) -2 Degree of AoA



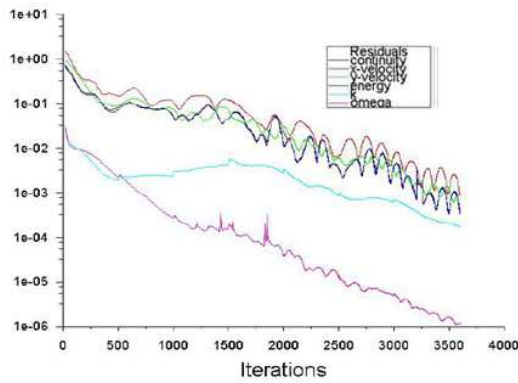
(d) 0 Degree of AoA



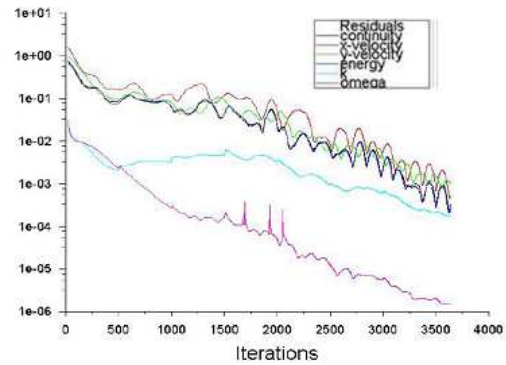
(e) 2 Degree of AoA



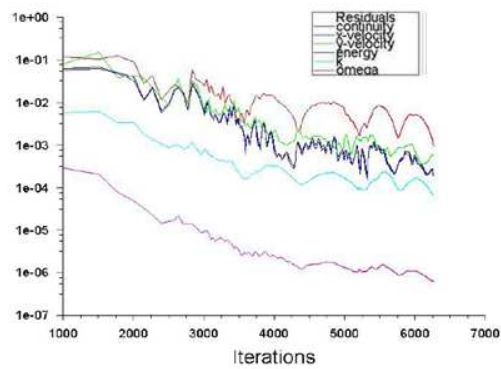
(f) 4 Degree of AoA



(g) 6 Degree of AoA

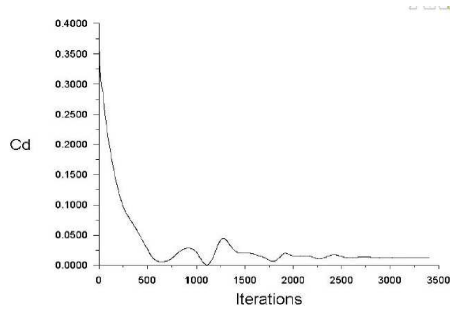


(h) 8 Degree of AoA

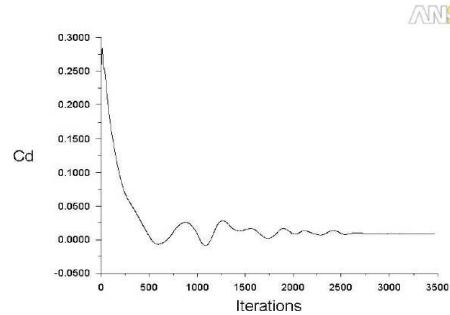


(i) 10 Degree of AoA

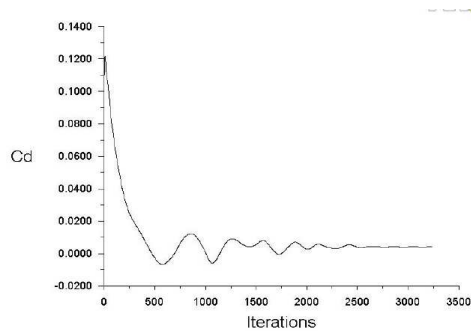
Figure 5(a)-(i) : Residual of Various Iterations Generated at -6 to 10 Degrees of AoA



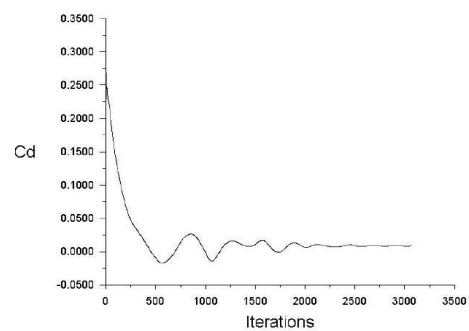
(a) -6 Degree of AoA



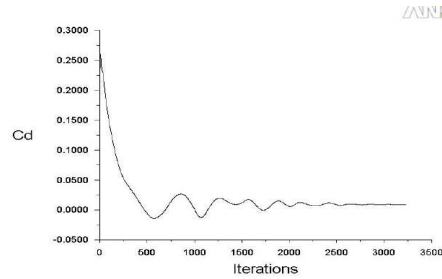
(b) -4 Degree of AoA



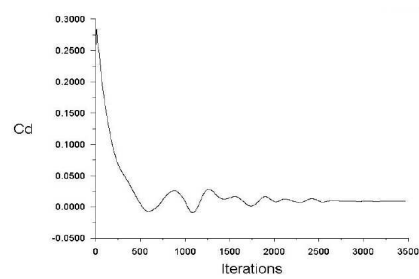
(c) -2 Degree of AoA



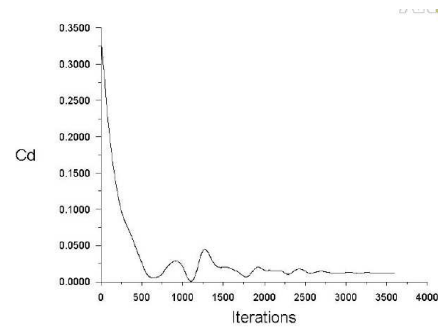
(d) 0 Degree of AoA



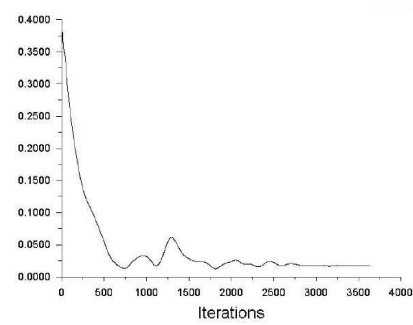
(e) 2 Degree of AoA



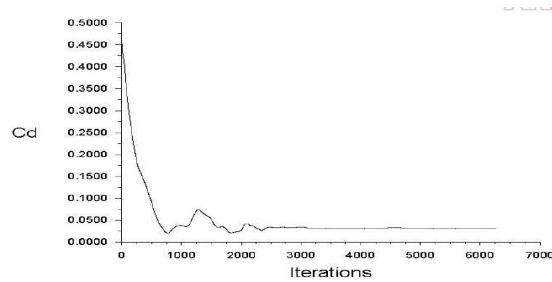
(f) 4 Degree of AoA



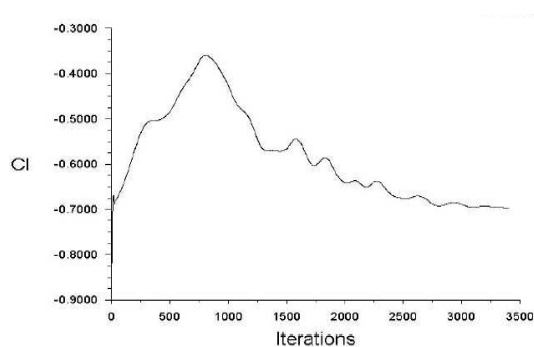
(g) 6 Degree of AoA



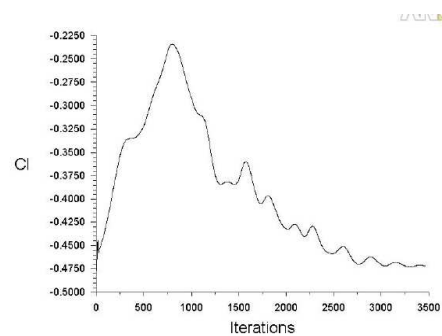
(h) 8 Degree of AoA



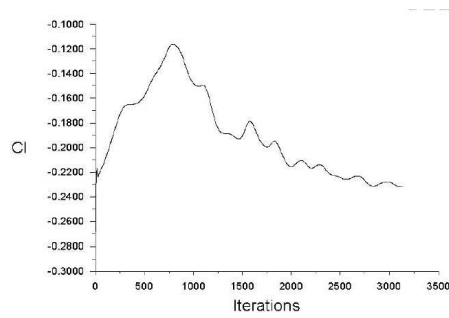
(i) 10 Degree of AoA

Figure 6(a)-(i) : Drag Convergence History
at -6 to 10 Degrees of AoA

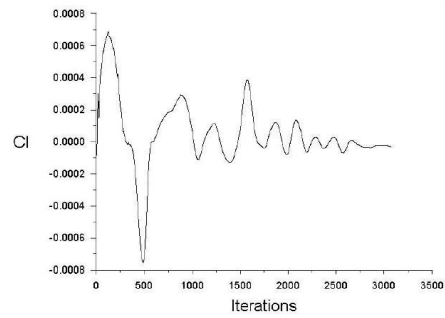
(a) -6 Degree of AoA



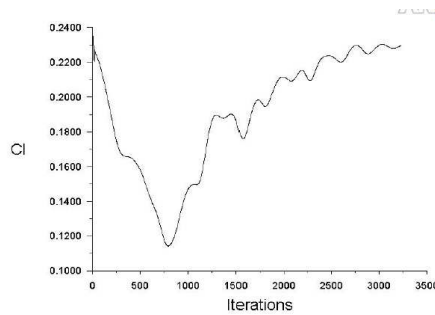
(b) -4 Degree of AoA



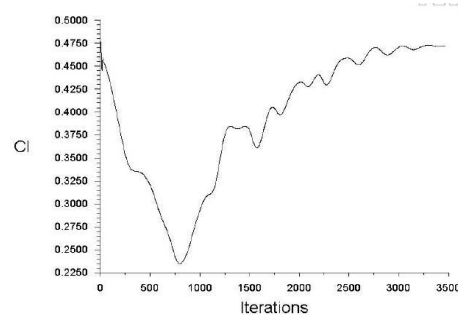
(c) -2 Degree of AoA



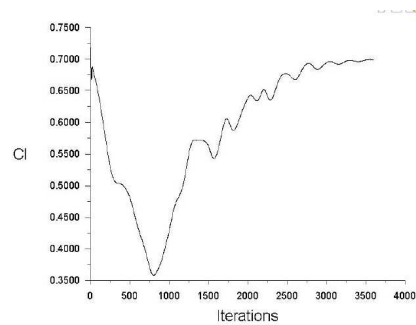
(d) 0 Degree of AoA



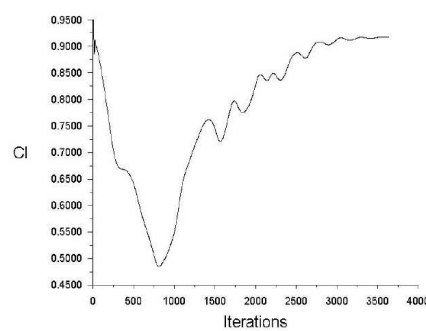
(e) 2 Degree of AoA



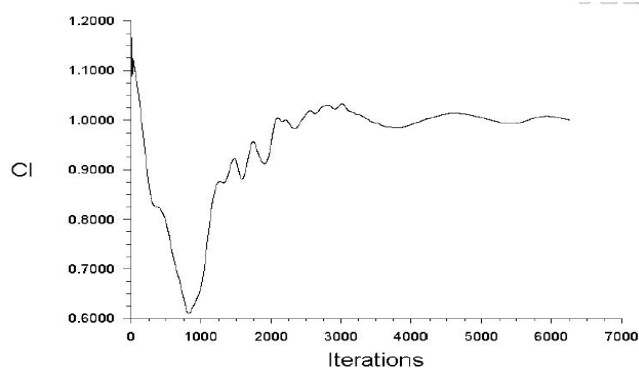
(f) 4 Degree of AoA



(g) 6 Degree of AoA

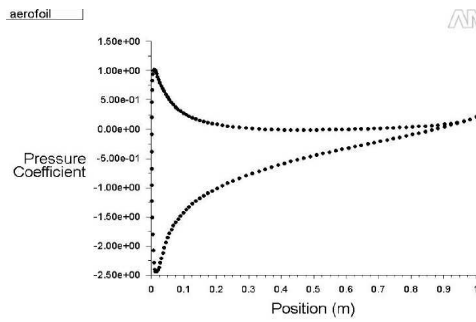


(h) 8 Degree of AoA

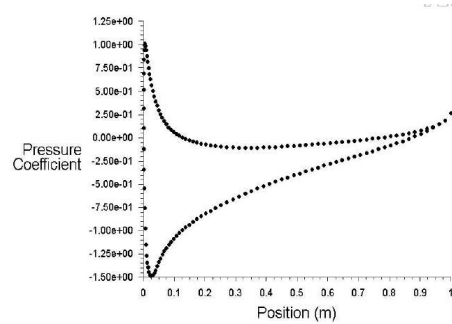


(i) 10 Degree of AoA

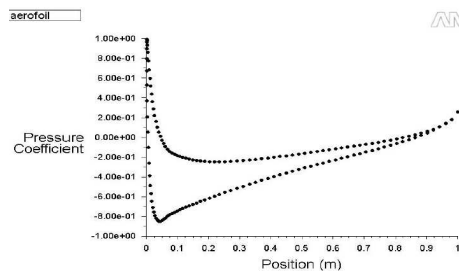
Figure 7(a)-(i) : Lift Convergence History at -6 to 10 Degrees of AOA



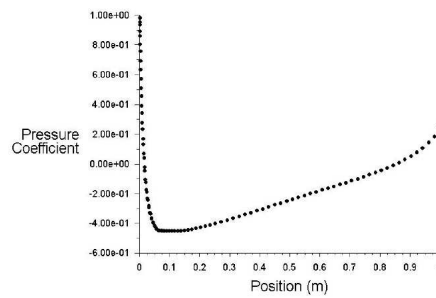
(a) -6 Degree of AoA



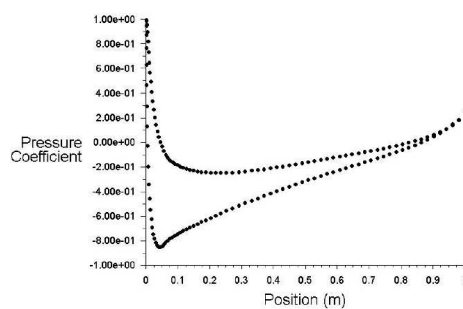
(b) -4 Degree of AoA



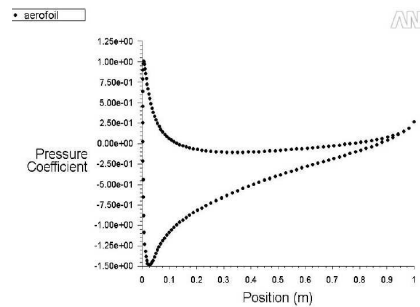
(c) -2 Degree of AoA



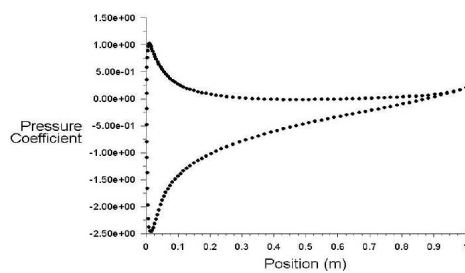
(d) 0 Degree of AoA



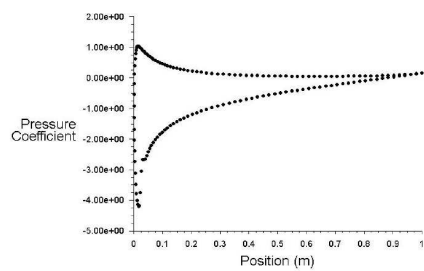
(e) 2 Degree of AoA



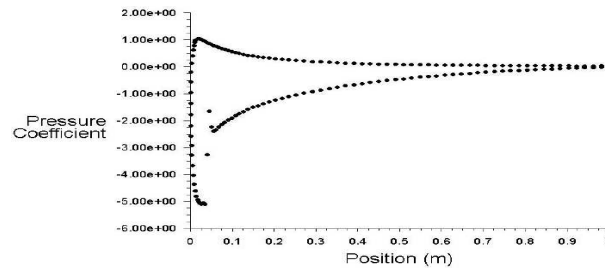
(f) 4 Degree of AoA



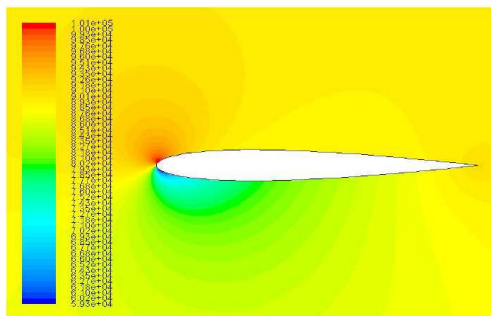
(g) 6 Degree of AoA



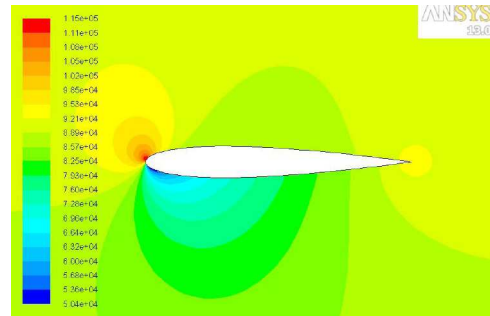
(h) 8 Degree of AoA



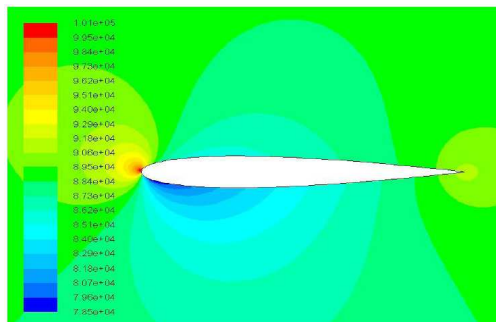
(i) 10 Degree of AoA
Figure 8(a)-(i) : Pressure Coefficient at -6 to 10 Degrees of AoA



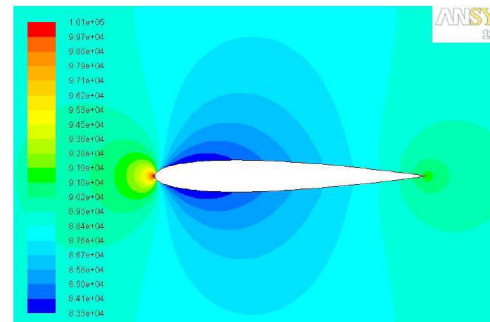
(a) -6 Degree of AoA



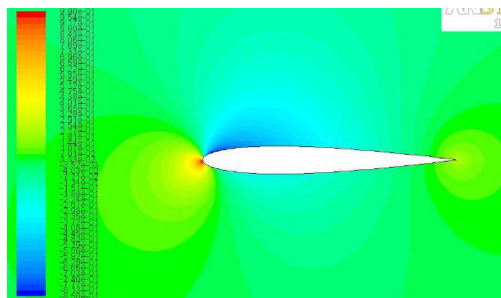
(b) -4 Degree of AoA



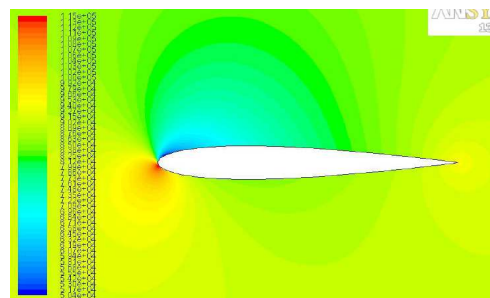
(c) -2 Degree of AoA



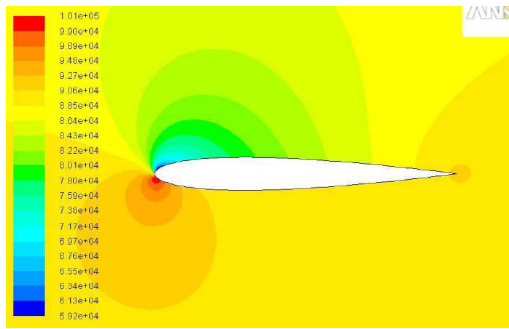
(d) 0 Degree of AoA



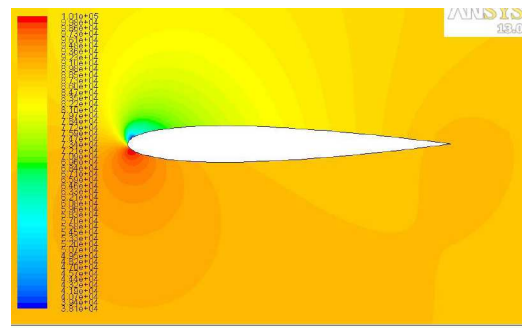
(e) 2 Degree of AoA



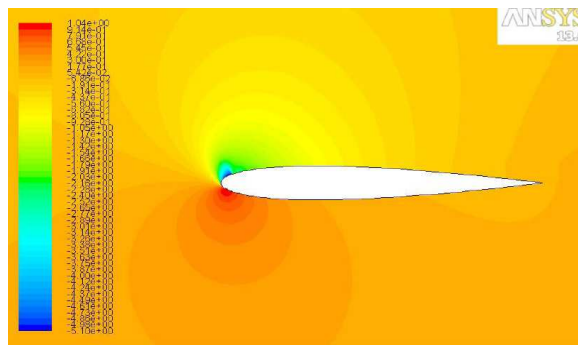
(f) 4 Degree of AoA



(g) 6 Degree of AoA

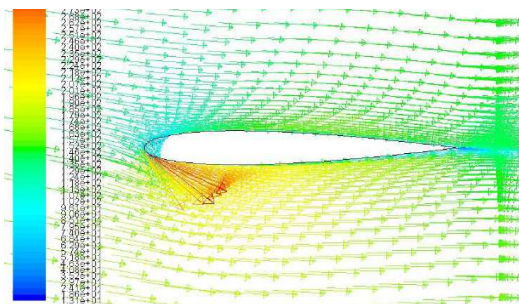


(h) 8 Degree of AoA

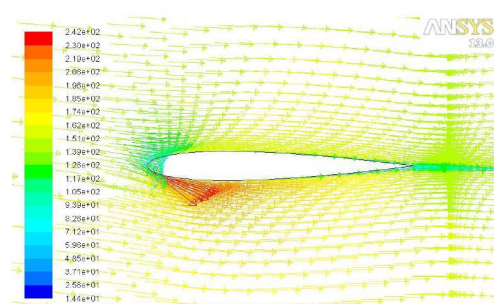


(i) 10 Degree of AoA

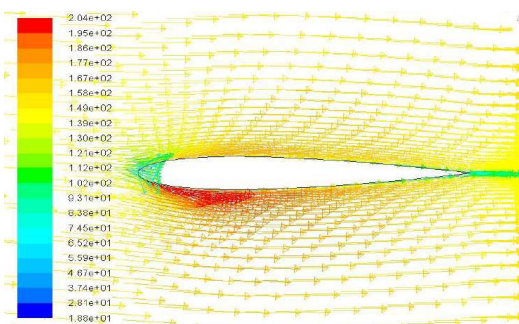
Figure 9(a)-(i) : Contours of Pressure Coefficient (Pascal) at -6 to 10 degrees of AoA



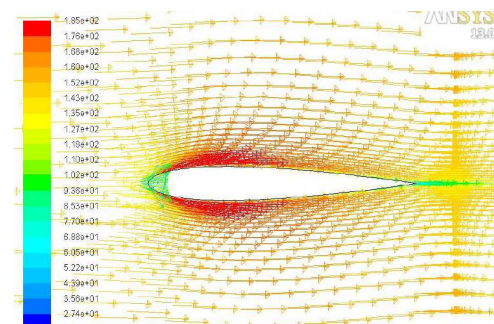
(a) -6 Degree of AoA



(b) -4 Degree of AoA



(c) -2 Degree of AoA



(d) 0 Degree of AoA

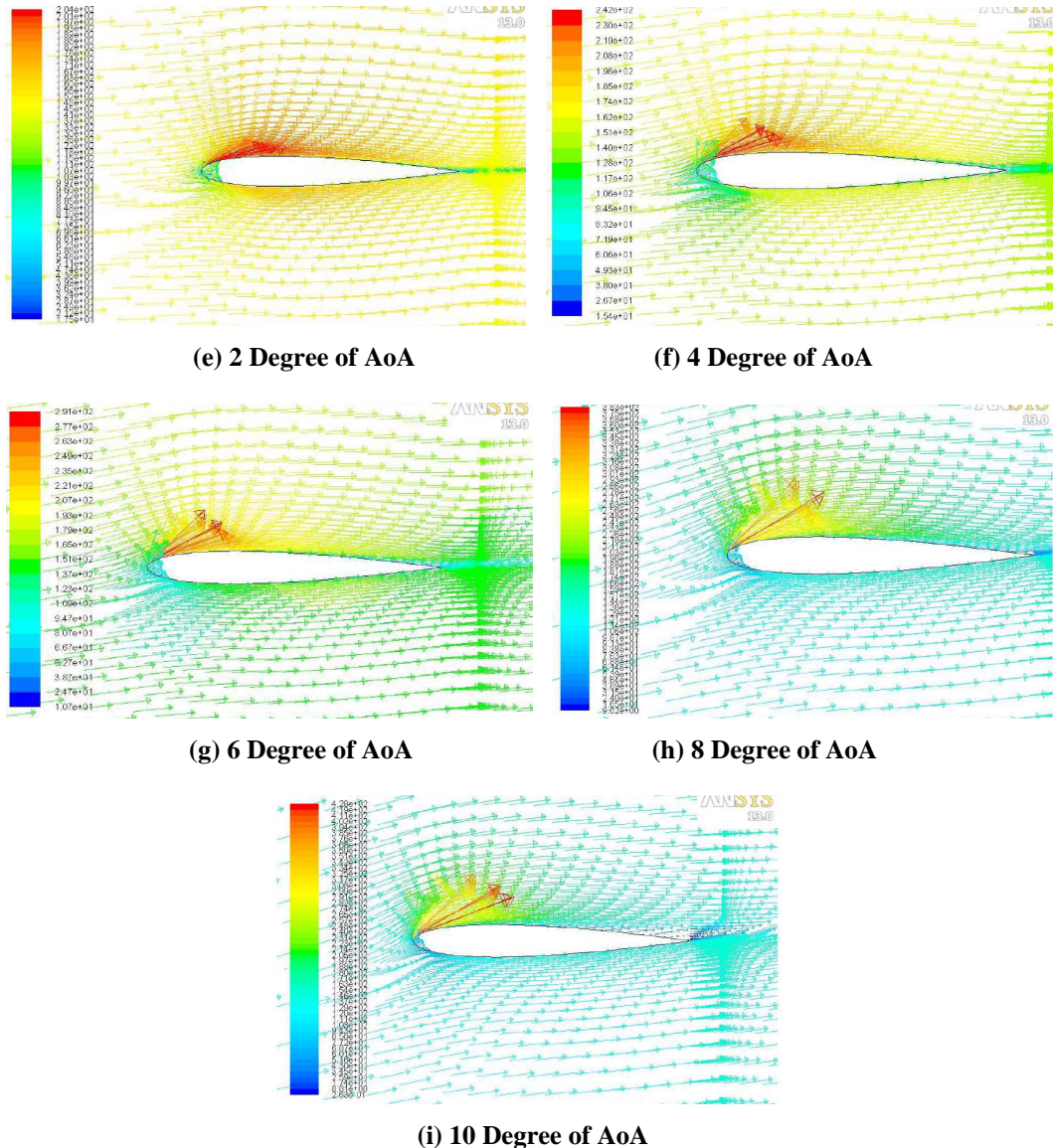


Figure 10(a)-(i) : Velocity Vector Colored by Velocity Magnitude (m/s) at -6 to 10 Degrees of AOA

CONCLUSIONS

In this research work, the precision of turbulence model of K- ω SST in the numerical analysis of flow over a NACA0012 airfoil is explored utilizing CFD. The consequences of K- ω SST turbulence show is intended for aviation applications and offers fine outcome for boundary layers being presented to the opposite weight. The drag coefficient anticipated by the turbulence model was more prominent than getting from the results. The computation comes about were given as takes after, coefficient drag and lift expanded with expanding approach.

In this investigation lift and drag exhibitions of NACA0012 airfoil were performed. A FLUENT program was utilized to numerical estimation. Numerical and trial comes about were analyzed. The estimation come about were given as takes over; Drag and lift performance expanded with expanding approach. Slow down begun 8 -degree angle attack. Lift coefficient diminished while the drag coefficient expanded. The ideal lift coefficient esteem was measured and figured at 10 degrees. The ideal airfoil execution was measured and ascertained at around 8 degrees numerical investigation were demonstrated a decent outcome and closeness.

REFERENCES

1. Pallavi Paturu, Gandhi Mallela, S. Jebarose Juliyana, A. Divya Sadhana & M. Komaleswarao, "Numerical Analysis of Flow Over Naca0012 at Fixed Mach Number using Computational Fluid Dynamics, *International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)* ISSN (P): 2249-6890; ISSN (E): 2249-8001, Vol. 7, Issue 6, Dec 2017, 213-222.
2. Douvi C. Eleni, Tsavalos I. Athanasios and Margaritis P. Dionissios, *Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil.*
3. Eleni c. Douvi, Athanasios I. Tsavalos, Dionissios P. Margaritis, *CFD calculations of the flow over a NACA0012 airfoil, Fluid Mechanics Laboratory (FML) University of Patras, Dept. of Mechanical Engineering and Aeronautics 26500 Patras, Greece.*
4. Karna S. Patel, Saumil B. Patel, Utsav B. Patel, Prof. Ankit P. Ahuja, *CFD Analysis of an Aerofoil, UVPCE, Ganpat University*
5. Mandapudi, Snigdha, Satya Sandeep Chaganti, And Swathi Gorle. "CFD Simulation of Flow Past Wing Body Junction: A 3-D Approach."
6. Connell, J. R., "The Spectrum of Wind Speed Fluctuations Encountered by a Rotating Blade of an Energy Conversion System," *Solar Energy*, Vol 29, No5, pp. 363–375, 1982.
7. Bertagnolio. F, Sørensen. N and Johansen. J, "Profile Catalogue for Airfoil Sections Based on 3D Computations" *Tech. Rep. Risø-R-1581(EN)*, RISØ National Laboratory, Roskilde, 2006.
8. S. Jebarose Juliyana, Dr. J. Udaya Prakash, K. Karthik, P. Pallavi and M. Saleem, *Design and analysis of NACA4420 wind Turbine aerofoil using CFD, International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 6, June 2017, pp. 403–410.*
9. Ranganayakulu Chennu, Pallavi Paturu, *Development of heat transfer coefficient and friction factor correlations for offset fins using CFD, International Journal of Numerical methods for Heat & Fluid flow, Volume 21, Issue 8, 2011, pp. 935-951.*